



Numerical Analysis on Flows Through Hydraulic Machinery(流体機械の内部流れの数値解析)

著者	呉 玉 林
号	1539
発行年	1994
URL	http://hdl.handle.net/10097/10346

氏	名	呉	玉	林
授 与 学 位	博 士	(工 学)		
学位授与年月日	平成 6 年 11 月 9 日			
学位授与の根拠法規	学位規則第 4 条第 2 項			
最 終 学 歴	昭 和 56 年 3 月			
	中国清華大学大学院水力機械学専攻修士課程修了			
学 位 論 文 題 目	Numerical Analysis on Flows Through Hydraulic Machinery (流体機械の内部流れの数値解析)			
論 文 審 査 委 員	東北大学教授 大場利三郎 東北大学教授 大宮司久明 東北大学教授 加藤 康司 東北大学助教授 井小萩利明			

論 文 内 容 要 旨

ENDFIELD

Chapter 1 Introduction

The flow through hydraulic machinery is one of the most complex flows in fluid-dynamic practice. In most instances, it is three-dimensional, with laminar-, transitional- and turbulent-flows and frequently separated-flows, and may be a single-phase- or a two-phase-flow. Computational Fluid Dynamics (CFD) has fostered a unified approach to turbomachinery analysis and design. In hydraulic machinery, traditionally, the design and the development has been based on a slight modification of existing components, and the performance assessment of them has relied on model testing. Computational methods based on the potential- or the inviscid-flow-approximation have been developed and applied to aid the optimum design of the turbine runners. However, they cannot represent the complex behavior of truly turbulent flows and are unable to predict the associated energy losses. In the past several years, therefore, the viscous/turbulent flow analysis has been applied to predict the flows through pump impellers and turbine runners.

In the present work, attention has been focused on the latest development of Computational Fluid Dynamics in hydraulic machinery. I have finished a series of researches in this field covering mainly the following aspects: The boundary element solution for potential flows through

h runners and other components; Three-dimensional boundary layer calculation, and the interaction between this calculation and the 3D potential solution; Incompressible turbulent flow computation; Incompressible liquid-particulate two-phase turbulence analysis.

Chapter 2 Boundary Element Solution for a Potential Flow in a Hydraulic Turbine

The guide passages of a turbine include the spiral casing, the stay vanes and the guide vanes. The flow simulation has been performed on this whole assembly by using 2D BEM. Under the irrotational assumption the 2D incompressible steady flow may be expressed in the Laplacian equation of the stream function. The calculation was carried out by using the boundary element method with linear triangular elements. In this calculation, the Kutta conditions are introduced to the BEM algebraic equations, then one of two unknown differentials in these conditions is eliminated from the BEM equations. This eliminated variable will be replaced by the tangential velocity at the outlet or by the stream functions on the vane surfaces. So that the solution of the BEM will be obtained without iterative processing to satisfy the Kutta conditions at each vane trailing edge. The 2D potential flow on the domain has been executed for an experimental apparatus of a spiral casing and guide vanes. Calculated results of pressure coefficients C_p are shown in Fig. 1 and Fig. 2. The agreement between the calculations and the experiments indicates the present method is capable to predict the potential flow from the inlet of the spiral casing to the outlet of the guide vanes in the Francis turbine.

For a relatively steady incompressible inviscid flow through a runner, under an irrotational absolute flow assumption, the absolute velocity potential Φ is introduced. Then, the governing equation of the 3D absolute potential flow becomes a Laplacian equation. The circulation in the upstream region of the runner is prescribed by a constant value to satisfy the irrotational condition, whereas the downstream circulation must satisfy the Kutta conditions. It is assumed that the extended boundaries downstream from the blade trailing edges are taken as flow surfaces, i. e., the vortex sheets shedding from the blade trailing edges. But their positions are still unknown before the calculation. The Kutta conditions are the dynamic conditions to determine the positions of these boundaries. So that the 3D potential solution through the runner is nonlinear. By using the BEM, the governing equation is transformed into an equivalent integral equation. Numerical computation is performed for a typical Francis turbine. Comparison of the calculated pressure distributions with the experimental ones are shown in Fig. 3. The calculated results shown in the figures agree quite well with the experiments.

Chapter 3 Three-Dimensional Boundary Layer Calculation and Its Application to Flow Simulation of Hydraulic Turbines

The direct mode and the inverse mode to solve the 3D incompressible turbulent boundary layer, developed in this study, have advantages of good accuracy and high computation speed owing to application of the semiorthogonal coordinate system, the turbulence model with consideration of anisotropy of Reynolds' stresses, the coordinate transformation technique and the Keller's box method. The present calculations have proved to be in good agreement with the experiments and to have an advantage in comparison with the previous calculations.

The inverse solution for the 3D boundary layer is applicable for calculation of the boundary layer with separation bubbles and small regions of boundary layer separations. But this inverse solution can not cover all separation flows. If the separation near the blade trailing edge is fully developed, the boundary layer theory will be invalid.

The 3D boundary layer direct solution and its interaction with the 3D potential flow calculation may be successfully used to predict the internal flow in turbine runners at the design operating condition, as shown in Fig. 4. The inverse solution of the 3D boundary layer on the blade surfaces and its interaction with the 3D potential flow calculation may be applied to analyze the internal flows in the runner at the off-design operating conditions, in which there are small separated regions or separated bubbles. Fig. 5 shows the pressure coefficient distributions along 4th streamlines S4 on the pressure side and the suction side of the runner blade at an off-design condition. The attack angle at the blade leading edge, $\alpha = -7.8^\circ$, is caused by an increase in the angular velocity. The negative attack angle and the adverse pressure distribution at the inlet of the blade pressure side cause a long separated bubble locating in the area of the pressure side leading part. Results of this technique show better agreement with measurements. But results of the interaction between the direct solution of boundary layer and the potential flow calculation does not show the separation.

Chapter 4 Incompressible Turbulent Flow Computation

The main numerical approaches for solving this problem by using the primitive variables include:

- (1) The pseudo-compressibility method.
- (2) The approximate factorization scheme (the fractional step scheme).
- (3) The successive pressure-velocity correction scheme.
- (4) The block-implicit finite difference method (BIFDM).

The successive pressure-velocity correction scheme is also called as the Semi-Implicit Method for Pressure Linked Equation (SIMPLE). To improve the rate of convergence, several new versions of this procedure were developed. Those are the SIMPLER (SIMPLE Revised), the SIMPLEST and the SIMPLEC (consistent SIMPLE). The SIMPLEC algorithm has been used in this calculation of turbulent flows and the liquid-particle two-phase turbulent flows though

a centrifugal impeller.

The Block Implicit Finite Difference Method (BIFDM) differs from the SIMPLE method, by employing a coupled solution of the momentum and the continuity equations. The coupling of these equations leads to a simultaneous updating on the pressures and the velocity-components by using the block-implicit finite difference scheme and a symmetrical coupled Gauss-Seidel (SCGS) technique. In the coupled approach, the continuity equation is used in its primitive form. Thus, the relation between the pressure and the velocities is implicitly retained.

The development of the BIFDM in this study is to use this technique to analyze turbulent flows in the complicated field, such as, in a 90° bend with square sections, by using the BFC system. Fig. 6 (a, b, c, d) indicate the pressure coefficient C_p distributions along sections $\theta = 0^\circ$, $\theta = 45^\circ$, $\theta = 71^\circ$ and $\theta = 90^\circ$, respectively. Good agreement between the calculated- and the measured-data even at the section $\theta = 90^\circ$. Another example is the 3D viscous flow in a rotating duct. Numerical analysis of the laminar rotating flows in such a passage with Reynolds number $Re = 10 - 1000$, and rotation number $Ro = 0 - 1$ have been carried out. Figs. 7 (a, b, c, d) show the typical results at $Re = 100$ and $Ro = 0.1$.

Chapter 5 Computation on Turbulent Dilute Liquid-Particle Flows by Using the $k-\epsilon-A_p$ Turbulence Model

The macroscopic continuum theory, particularly, the two-fluid model is used to simulate the liquid-particulate two-phase flows. In this model, the dispersed phase is treated as a pseudo-fluid. The $k-\epsilon-A_p$ turbulence model is adopted to describe the turbulent flow process. This model is the $k-\epsilon$ turbulence model for liquid phase combined with the algebraic turbulence model A_p for particulate phase. The finite volume numerical method is presented for the simulation. A mesh in body-fitted coordinates is generated for the computation. In detail, the SIMPLEC algorithm is utilized for solving the governing equations. Fig. 8 show the relative velocity distributions through a centrifugal impeller at the liquid-particle two-phase condition. The velocity field of the particulate phase is different from that of the liquid phase in the centrifugal impeller, especially near the outlet ring. Relative velocities of particles are not slowed down near the outlet. At the dilute concentration, existence of particle scarcely affects the liquid-phase flow in the impeller. But it makes the pressure at the impeller outlet a little lower than that of the single-phase flow. The larger the particle diameter is, the more the impeller outlet pressure decreases. It also makes the maximum value of the turbulent kinetic energy be smaller than that at single flow condition.

Chapter 6 Conclusions

The present work is aimed at developing numerical analyses on hydraulic machinery. Some

of the important conclusions obtained in this paper are summarized in the following:

(1) In this dissertation, a 2D simplified model- and a 2D potential flow-calculation by using the BEM for the whole field ahead of a Francis runner, including the spiral casing, the stay vanes and the guide vanes, provided for good agreement between the calculations and the experiments.

(2) For Francis turbines and pumps, the quasi-3D Euler methods have the advantages of taking account the rotating effects, but without covering the 3D effects. On the other hand, the fully 3D potential flow codes yield the 3D effects, but only when there are irrotationality generally. As the flows through Francis runners and centrifugal impellers are rotational, a 3D potential flow analysis needs a superpositional technique to satisfy the Kutta condition. In this study, the nonlinear technique considering the vortex sheets downstream from the runner/impeller, has been applied by using the BEM. This fully 3D potential solution has proved to be effective and applicable to a wide range of design conditions including the nonuniform blade loading.

(3) To solve viscous internal flows through runner/impellers in a practical way, the 3D inviscid-viscous interaction technique between 3D potential flow and 3D turbulent boundary layer is developed in this dissertation. Especially, by using the inverse solution technique of 3D boundary layer. The interaction can be applied to predict the internal flows through runners/impellers at the off-design conditions. While the interaction between the potential flow and the 3D boundary layer direct solution may only be used to predict the internal flows through runners/impellers at the design condition.

(4) The incompressible turbulent flow computation, using the Reynolds-averaged N-S equations in the primitive variables combined with the turbulence model and the BFC technique, is now becoming more feasible even for complex geometrical computational domains. In this study, one of the pressure-velocity correction algorithms, the SIMPLEC has been used to predict the internal turbulent flow through a centrifugal impeller. This algorithm is reliable and has advantages of good convergence and stability.

We also have developed a block-coupled solution for the N-S equations. The block-implicit finite difference method (BIFDM) has been successfully extended to calculate the complex 3D incompressible viscous/turbulent flows of curved or rotating ducts by using the BFC technique and the $k-\varepsilon$ turbulence model.

(5) In order to simulate the liquid-particulate two-phase flow, a two-phase turbulence model, i.e., the $k-\varepsilon-A_p$ model and the computer algorithm based on the SIMPLEC method, have been developed in this study. The calculated results of the two-phase flow through a centrifugal impeller at dilute concentrations have indicated that this turbulence model is able to predict essential features of the complex two-phase flow. The basic phenomenon from

the computation is proved by the experimental results in particle motion and particle number distribution in a centrifugal impeller. It is found that the velocity of solid particle at the outlet area of the impeller, especially near the blade pressure surface is not slowed down appreciably, so that the slip velocity between the solid phase and the liquid phase is positive at this part of the flow field.

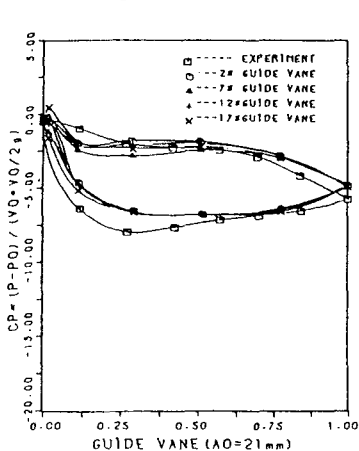


Fig. 1 Pressure on Guide Vanes

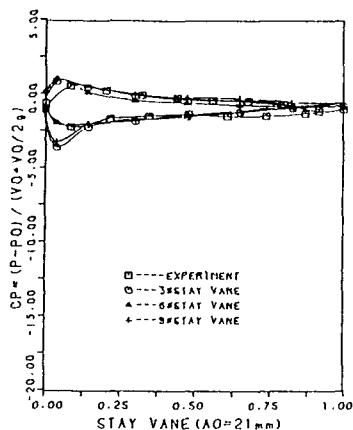
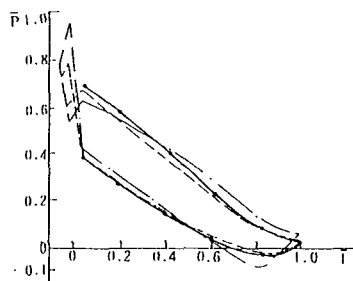


Fig. 2 Pressure on Stay Vanes



(a) On S4 Streamline

Fig. 3 Pressure Distribution on Blade at Design Condition

— measured - - - potential flow
- · - interaction between potential and direct boundary layer

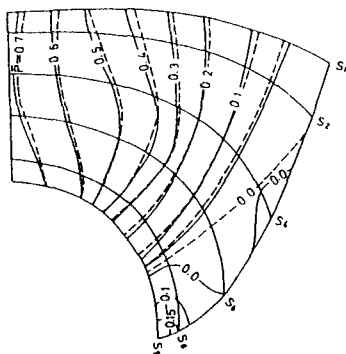
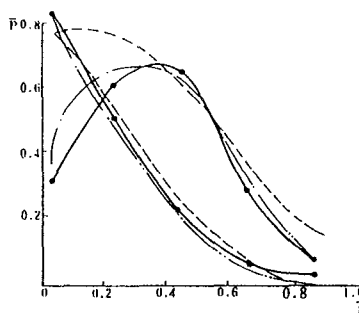


Fig. 4 Equi-Pressure Lines on Blade Pressure Side

— measured
— present calculated



(Pressure Distribution)

Fig. 5 Calculated Results at $n_{11} = 1.27n_{110}$ on S4 streamline

— measured — IVI (direct mode)
- - - IVI (inverse mode)

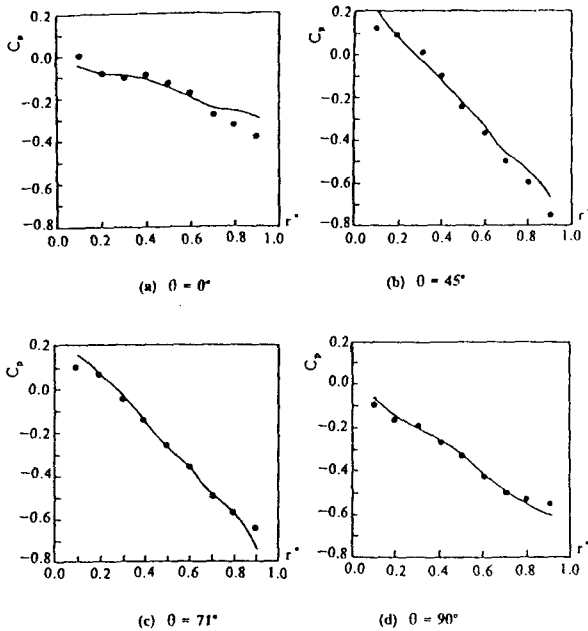
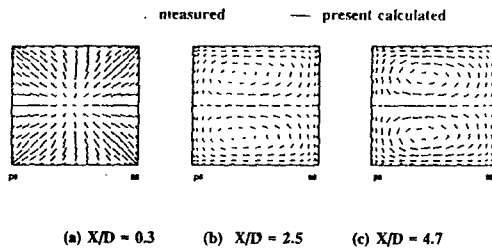


Fig. 6 Pressure Comparison



Typical Secondary Flows

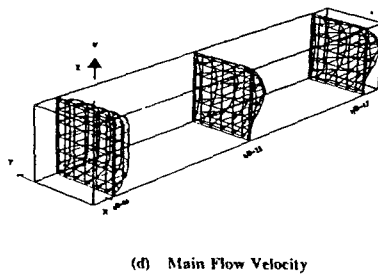
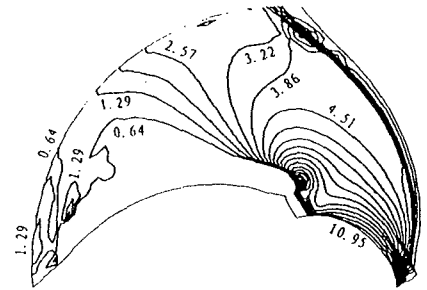
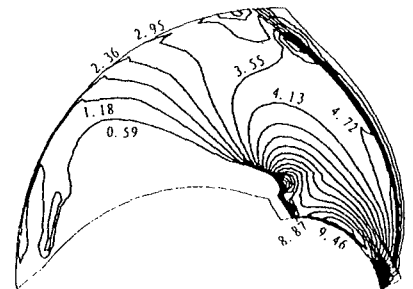


Fig. 7 Main Flows and Typical Secondary Flows in a Rotating Duct
($R_o = 100$, $R_i = 0.1$)



(a) Water Flow ($W_{max} = 12.24$ m/s)



(b) Particle Flow ($W_p = 11.13$ m/s)

Fig. 8 Relative Velocity at Condition

$u_{t1} = 6$ m/s, $n = 1450$ rpm, $C_p = 0.1$, $d_i = 0.1$ mm and $\rho_i = 2.6$ g/cm³

審 査 結 果 の 要 旨

水車等の流体機械の更なる性能向上を図るため、現在、その流体力学的設計においては、複雑な内部流れの数値シミュレーション手法の導入が不可欠とされている。本論文は、このような要請を踏まえて、境界要素法によるポテンシャル解法、三次元乱流境界層の直接法および逆解法、非圧縮性乱流の数値解法ならびに固液二相流の数値解法を提示し、その有用性を数値流体力学的に明らかにしたもので、全編6章よりなる。

第1章は緒論である。

第2章では、境界要素法を用いた水車内部流れのポテンシャル解法を提示している。まず、ステータおよびガイドベーンを含む渦巻ケーシング内部流れについて、流線関数を未知変数とするラプラス方程式に、二次元境界要素法を適用し、Kuttaの条件を考慮することにより、反復計算を必要としない効率的な解法となること、ステータおよびガイドベーンの圧力分布は実験値とも良く一致することが示された。次いで、フランスス水車ランナー内部流れについて、絶対速度のポテンシャルを未知変数とするラプラス方程式に三次元境界要素法を適用した非線形解法を展開した。翼形後縁からの渦層の流出とKuttaの条件を考慮した計算結果は、実験値との一致も良いので、境界要素法を用いた本解法の妥当性が示されたと言える。

第3章では、非等方代数渦粘性モデルを組み込んだ三次元乱流境界層方程式にBox法を適用し、ポテンシャル外部流を境界条件とする直接解法および境界層の排除効果を考慮して粘性-非粘性干渉を評価する逆解法を提示している。これらより、直接解法では境界層はく離のない設計状態での水車ランナーの境界層特性を良好に予測できること、非設計状態に対しては、はく離が予測できる逆解法が極めて有効となることなどが明らかにされ、本境界層計算手法の有用性が検証された。

第4章では、非圧縮性乱流の数値計算法を確立するために、一般曲線座標系のナビエ・ストークス方程式に標準 $k-\epsilon$ 乱流モデルを組み込んだ2種類の計算コードの開発を行っている。これらは、SIMPLEC法を適用した有限体積法とガウス・ザイデル反復法を用い圧力・速度の同時陰的緩和を行うBIFDM法に基づく計算コードで、前者は二次元の後方段付き流路と遠心ポンプの内部流れに、後者は 180° および 90° 矩形曲管と三次元回転ダクトの内部流れに適用され、乱流特性や二次流れの計算にも有効であることを実証した。

第5章では、遠心ポンプ羽根車内の低濃度固液二相流動特性を解明するため、固体粒子の軌跡と数密度分布を実験的に明らかにするとともに、固体粒子による乱流エネルギーの生産を考慮した $k-\epsilon-A_p$ 二相乱流モデルを有限体積法に組み込んだ計算コードを開発し、数値計算によりポンプ出口での粒子は減速せず、粒子径が大なるほど出口圧力および最大乱流エネルギーが単相流と比べて減少することなどが見い出された。これは重要な知見である。

第6章は結論である。

以上要するに本論文は、流体機械の内部流れの各種数値解析法を発展的に提示し、それらの計算コードの開発と有用性を明らかにしたもので、数値流体力学ならびに流体機械学の発展に寄与するところが少なくない。

よって、本論文は博士（工学）の学位論文として合格と認める。